

Tutorial: Coal Combustion with Eddy Break Up (EBU) Model

Introduction

The purpose of this tutorial is to provide guidelines and recommendations for setting up and solving a coal combustion case using coal calculator.

This tutorial demonstrates how to do the following:

- Set up and solve a coal combustion case.
- Solve the case using appropriate solver settings.
- Postprocess the resulting data.

Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1 from the ANSYS FLUENT 14.5 Tutorial Guide, and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

If you have not used EBU model before, it would be helpful to first refer to the ANSYS FLUENT 14.5 User’s Guide and the ANSYS FLUENT 14.5 Tutorial Guide.

Problem Description

A 3D cutaway of the furnace is shown in Figure 1. Two annular inlets on the left-hand side and a circular outlet on the right-hand side are visible. Only one quarter of this geometry is modeled due to symmetry. The inner annular inlet has inner and outer radii of 0.055 m and 0.067 m respectively. The outer annular inlet has inner and outer radii of 0.07 m and 0.117 m respectively. The outlet radius is 0.425 m.

Coal and carrier air enter the combustion chamber through the inner annular region. Hot, swirling, secondary air enters through the outer annular region. Combustion takes place and the products exit at the pressure outlet.
Preparation

1. Copy the files, coal-ebu.msh.gz and coal-ebu.c, to your working folder.

2. Use FLUENT Launcher to start the (3D) version of ANSYS FLUENT.

3. Enable Double Precision in the Options list.

4. Click the Environment tab and make sure that Setup Compilation Environment for UDF is enabled.

   *The path to the .bat file which is required to compile the UDF will be displayed as soon as you enable Setup Compilation Environment for UDF.*

   *If the Environment tab does not appear in the FLUENT Launcher dialog box by default, click the Show More Options button to view the additional settings.*

Setup and Solution

Step 1: Mesh

1. Read the mesh file (coal-ebu.msh.gz).

2. Change the Periodic Type of periodic to Rotational.

   ![Boundary Conditions](image)

   (a) Select Rotational in the Periodic Type list.

   (b) Click OK to close the Periodic dialog box.
Step 2: General Settings

1. Check the mesh.

2. Display the mesh.

(a) Select all the surfaces from the Surfaces selection list.
(b) Click Display and close the Mesh Display dialog box.

Figure 2: Mesh Display

Step 3: Models

1. Enable the Energy Equation.

2. Select the standard k-epsilon (2 eqn) turbulence model.
3. Select the Discrete Ordinates model.

   ![Radiation Model dialog box](image)

   (a) Select Discrete Ordinates (DO) in the Model list.
   
   The Radiation Model dialog box expands to show the related inputs.
   
   (b) Set Energy Iterations per Radiation Iteration to 1.
   
   (c) Set Theta Divisions and Phi Divisions to 4 in the Angular Discretization group box.
   
   (d) Set Theta Pixels and Phi Pixels to 3 in the Angular Discretization group box.
   
   (e) Click OK to close the Radiation Model dialog box.
   
   Click OK in the Information dialog box.

4. Select the Species Transport model.

   (a) Select Species Transport in the Model list.
   
   (b) Enable Volumetric in the Reactions list.
   
   (c) Select Eddy Dissipation from the Turbulence-Chemistry Interaction list.
   
   (d) Click on Coal Calculator....
**Note:** Coal calculator is a tool to convert available inputs like proximate and ultimate analysis, and heating value of a fuel, into simulation inputs like stoichiometric coefficients of volatile break up or combustion reaction, standard state enthalpy of volatile species, volatile and combustible fractions of combusting material, and so on. This conversion is quick and accurate and therefore, manual calculations and errors are avoided.

i. Specify **Proximate Analysis** as shown in the table.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volatile</td>
<td>0.51</td>
</tr>
<tr>
<td>Fixed Carbon</td>
<td>0.34</td>
</tr>
<tr>
<td>Ash</td>
<td>0.08</td>
</tr>
<tr>
<td>Moisture</td>
<td>0.07</td>
</tr>
</tbody>
</table>

Table 1: Proximate Analysis
Coal Combustion with Eddy Break Up (EBU) Model

ii. Specify Ultimate Analysis (DAF) as shown in the table.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>0.80</td>
</tr>
<tr>
<td>H</td>
<td>0.05</td>
</tr>
<tr>
<td>O</td>
<td>0.13</td>
</tr>
<tr>
<td>N</td>
<td>0.02</td>
</tr>
</tbody>
</table>

Table 2: Ultimate Analysis (DAF)

iii. Select Two-Step Reaction from the group of Mechanism.

iv. Enable Wet Combustion from the Options group box.

v. Enter 3.058e+7 for Coal As-Received HCV.

vi. Enter 50 for Volatile Molecular Weight.

vii. Enter 0.775 for CO/CO2 Split in Reaction-1 Products.

viii. Enter 1.1 for High Temperature Volatile Yield.

ix. Enter 1200 for Coal Dry Density.

Note: CO/CO2 split is the fraction of CO in the volatile combustion reaction.

x. Click OK and close the Coal Calculator dialog box.

Note: A Warning dialog box is displayed to inform you that a number of mixture materials, their properties and reactions have been set up. From the Warning dialog box note down the values set for Volatile N mass fraction and Char N mass fraction. You will be using these values later (Step 13). Click OK to close the Warning dialog box.

(e) Click OK to close the Information dialog box, which informs about the change in material properties and methods.

(f) Click OK to close the Species Model dialog box.

Click OK in the Information dialog box.

5. Set the Discrete Phase model.

   ![Models](Discrete Phase) Edit...

(a) Enter 40000 for Max. Number of Steps.

(b) Enable Specify Length Scale and enter 0.0025 m for Length Scale.

(c) Click OK to close the Discrete Phase Model dialog box.
Step 4: Injections

1. Define 9 injections from surface v-1.

   (a) Click the Create button to open the Set Injection Properties dialog box.
   (b) Select surface from the Injection Type drop-down list.
   (c) Select v-1 from the list of Release From Surfaces.
   (d) Select Combusting from the Particle Type group box.
   (e) In the Point Properties tab, enter 23.11 for Z-Velocity and 343 for Temperature.
   (f) Enter 0.00018264 for Total Flow Rate.
   (g) Click the Turbulent Dispersion tab and enable Discrete Random Walk Model.
      i. Enter 10 for Number of Tries.
   (h) Click the Wet Combustion tab and enable Wet Combustion Model.
(i) Select h20 from the Evaporating Species drop-down list. 
*Liquid material and liquid fraction are automatically added.*

(j) Click OK to set injection-0.

(k) The properties specific to each injection are shown in the table:

<table>
<thead>
<tr>
<th>Injection Name</th>
<th>Diameter (m)</th>
<th>Flow Rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>injection-0</td>
<td>1e-6</td>
<td>0.00018264</td>
</tr>
<tr>
<td>injection-1</td>
<td>5e-6</td>
<td>0.00073056</td>
</tr>
<tr>
<td>injection-2</td>
<td>1e-5</td>
<td>0.00127848</td>
</tr>
<tr>
<td>injection-3</td>
<td>2.5e-5</td>
<td>0.00438336</td>
</tr>
<tr>
<td>injection-4</td>
<td>5e-5</td>
<td>0.00584448</td>
</tr>
<tr>
<td>injection-5</td>
<td>7.5e-5</td>
<td>0.00347016</td>
</tr>
<tr>
<td>injection-6</td>
<td>0.0001</td>
<td>0.00146112</td>
</tr>
<tr>
<td>injection-7</td>
<td>0.0002</td>
<td>0.00073056</td>
</tr>
<tr>
<td>injection-8</td>
<td>0.0003</td>
<td>0.00018264</td>
</tr>
</tbody>
</table>

Table 3: Specific Injection Properties

(l) Retain the default values for the other parameters.

(m) Close the Injections dialog box.

Step 5: Materials

1. Modify the properties for the coal-volatiles-air mixture.

(a) Set the physical properties for following parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal Conductivity</td>
<td>polynomial. The first and second temperature coefficients are 0.01006 and 5.413e-5 respectively.</td>
</tr>
<tr>
<td>Viscosity</td>
<td>polynomial. The first and second temperature coefficients are 9.18e-6 and 3.161e-8 respectively.</td>
</tr>
<tr>
<td>Absorption Coefficient</td>
<td>wsggm-domain-based</td>
</tr>
<tr>
<td>Scattering Coefficient</td>
<td>constant with a value of 0.5</td>
</tr>
</tbody>
</table>

Table 4: Properties for Parameters

(b) Click Change/Create.
2. Set the properties for the combusting particle coal-particle.

![Materials ➔ coal-particle ➔ Create/Edit...]

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cp</td>
<td>1100</td>
</tr>
<tr>
<td>Vaporization Temperature</td>
<td>400</td>
</tr>
<tr>
<td>Binary Diffusivity</td>
<td>3e-5</td>
</tr>
<tr>
<td>Swelling Coefficient</td>
<td>2</td>
</tr>
<tr>
<td>React. Heat Fraction Absorbed by Solid</td>
<td>0</td>
</tr>
<tr>
<td>Devolatilization Model</td>
<td>kinetics/diffusion-limited</td>
</tr>
<tr>
<td></td>
<td>Mass Diffusion-Limited Rate Constant = 5e-12</td>
</tr>
<tr>
<td></td>
<td>Kinetics-Limited Rate Pre-Exponential Factor = 6.7</td>
</tr>
<tr>
<td></td>
<td>Kinetics-Limited Rate Activation Energy = 1.138e8</td>
</tr>
</tbody>
</table>

Table 5: Combusting Particle Material Properties

Vaporization temperature of coal is 773 K. To start the reactions, lower the temperature to 343 K and once the flame shape is obtained, it will be changed to the original value.

3. Set the properties for the droplet particle.

![Materials ➔ water-liquid ➔ Create/Edit...]

(a) Select piecewise-linear from the Cp(Specific Heat) drop-down list and retain the default values.

(b) Enter 360 for Vaporization Temperature.

(c) Select convection/diffusion controlled from the Vaporization Model drop-down list.

(d) Click Change/Create.


Step 6: Compiling the Interpreted User Defined Functions (UDFs)

These functions will be used later to set the boundary conditions. For more information on interpreted UDFs, refer to the ANSYS FLUENT 14.5 UDF Manual.

Define ➔ User-Defined ➔ Functions ➔ Interpreted...

1. Enter the name of the C function (coal-ebu.c) for Source File Name.

2. Specify the C preprocessor to be used in the CPP Command Name.

Keep the default Stack Size setting of 10000, unless the number of local variables in your function will cause the stack to overflow. In this case, set the Stack Size to a number that is greater than the number of local variables used.
3. Select the Use Contributed CPP option if you want to use the preprocessor supplied by ANSYS, Inc., instead of using your own.

4. Click Interpret and close the Interpreted UDFs dialog box. 

   *In case there are errors while interpreting, keep the dialog box open and continue debugging and interpreting simultaneously until no more errors are reported.*

**Step 7: Boundary Conditions**

1. Set the boundary conditions for v-1 as specified in Table 6.

   ![Boundary Conditions](v-1)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity Magnitude</td>
<td>23.11 m/s</td>
</tr>
<tr>
<td>Specification Method</td>
<td>Intensity and Hydraulic Diameter</td>
</tr>
<tr>
<td>Turbulence Intensity</td>
<td>10%</td>
</tr>
<tr>
<td>Hydraulic Diameter</td>
<td>0.013 m</td>
</tr>
<tr>
<td>Temperature</td>
<td>343 K</td>
</tr>
<tr>
<td>Internal Emissivity</td>
<td>1</td>
</tr>
<tr>
<td>Species Mass Fractions</td>
<td>( o_2 = 0.2315 )</td>
</tr>
<tr>
<td>Discrete Phase BC Type</td>
<td>escape</td>
</tr>
</tbody>
</table>

   Table 6: Boundary Conditions for v-1

2. Set the boundary conditions for v-2 as specified in Table 7.

   ![Boundary Conditions](v-2)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity Specification Method</td>
<td>Components</td>
</tr>
<tr>
<td>Coordinate System</td>
<td>Cylindrical (Radial, Tangential, Axial)</td>
</tr>
<tr>
<td>Radial-Velocity</td>
<td>0</td>
</tr>
<tr>
<td>Tangential-Velocity</td>
<td>udf vinlet2wvel</td>
</tr>
<tr>
<td>Axial-Velocity</td>
<td>udf vinlet2uvel</td>
</tr>
<tr>
<td>Specification Method</td>
<td>Intensity and Hydraulic Diameter</td>
</tr>
<tr>
<td>Turbulence Intensity</td>
<td>12 %</td>
</tr>
<tr>
<td>Hydraulic Diameter</td>
<td>0.047 m</td>
</tr>
<tr>
<td>Temperature</td>
<td>573 K</td>
</tr>
<tr>
<td>Species Mass Fractions</td>
<td>( o_2 = 0.2315 )</td>
</tr>
</tbody>
</table>

   Table 7: Boundary Conditions for v-2

3. Set the boundary conditions for p-1 as specified in Table 8.

   ![Boundary Conditions](p-1)

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Species Mass Fractions</td>
<td>( o_2 = 0.2315 )</td>
</tr>
</tbody>
</table>

   Table 8: Boundary Conditions for p-1
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gauge Pressure</td>
<td>0</td>
</tr>
<tr>
<td>Specification Method</td>
<td>Intensity and Hydraulic Diameter</td>
</tr>
<tr>
<td>Backflow Turbulence Intensity</td>
<td>10%</td>
</tr>
<tr>
<td>Backflow Hydraulic Diameter</td>
<td>1 m</td>
</tr>
<tr>
<td>Backflow Total Temperature</td>
<td>1000 K</td>
</tr>
<tr>
<td>Species Mass Fractions</td>
<td>( \text{o}_2 = 0.2315 )</td>
</tr>
</tbody>
</table>

Table 8: Boundary Conditions for p-1

4. Set the boundary conditions for the wall zones. The Temperature and Internal Emissivity are specified in Table 9.

<table>
<thead>
<tr>
<th>Zone</th>
<th>Temperature</th>
<th>Internal Emissivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>w-1</td>
<td>343</td>
<td>0.6</td>
</tr>
<tr>
<td>w-2</td>
<td>573</td>
<td>0.6</td>
</tr>
<tr>
<td>w-3</td>
<td>873</td>
<td>0.6</td>
</tr>
<tr>
<td>w-4</td>
<td>1273</td>
<td>0.5</td>
</tr>
<tr>
<td>w-5</td>
<td>udf wall5temp</td>
<td>0.5</td>
</tr>
<tr>
<td>w-6</td>
<td>udf wall6temp</td>
<td>0.5</td>
</tr>
<tr>
<td>w-7</td>
<td>udf wall7temp</td>
<td>0.5</td>
</tr>
<tr>
<td>w-8</td>
<td>1323</td>
<td>0.5</td>
</tr>
<tr>
<td>w-9</td>
<td>1073</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Table 9: Wall Boundary Conditions

**Step 8: Non-Reacting Flow Solution**

1. Disable Volumetric reactions.

   Models → Species → Edit...

2. Set Number of Continuous Phase Iterations per DPM Iteration to 0 to avoid injection of particles in this step.

   Models → Discrete Phase → Edit...

3. Set convergence to none.

   Monitors (Residuals) → Edit...
   (a) Select none from Convergence Criterion drop-down list.

4. Set the solution controls.

   Solution Methods
   (a) Select Coupled from the Scheme drop-down list.
Coal Combustion with Eddy Break Up (EBU) Model

5. Set the under-relaxation factors.
   - Solution Controls
     - (a) Enter 50 for Flow Courant Number.
     - (b) Enable Set All Species URFs Together.
     - (c) Click on Equations... and de-select Discrete Ordinates from the list of Equations.

6. Initialize the solution.
   - Solution Initialization
     - (a) Select Standard Initialization from the Initialization Methods group box.
     - (b) Enter 100 for Turbulent Dissipation Rate.
     - (c) Enter 0.2315 for \( \text{O}_2 \).
     - (d) Enter 1355 for Temperature.
     - (e) Click Initialize.

7. Start the calculation by requesting 100 iterations.
   - Run Calculation

8. Save the case and data files coal-ebu-cold.cas/dat.gz.

Step 9: Initiate Reacting Flow Solution

1. Change the settings in the Discrete Phase Model dialog box.
   - Models ➔ Discrete Phase ➔ Edit...
     - (a) Set the Number of Continuous Phase Iterations per DPM Iteration to 1.
     - (b) In the Physical Models tab enable Pressure Dependent Boiling.
       - Click OK in the Information dialog box.
     - (c) Click the Numerics tab.
       - i. Enable the term Enable Node Based Averaging in the Averaging group box.
       - ii. Ensure that Average DPM Source Terms is enabled.
       - iii. Enter 6 for Gaussian Factor in the Kernel Settings group box.
       - iv. Click OK to close the Discrete Phase Model dialog box.

2. Enable Volumetric reactions.
   - Models ➔ Species ➔ Edit...

3. Patch high temperature and product species mass fractions in reaction zone.
   - Adapt ➔ Region...
(a) Select Cylinder from the Shapes list.
(b) Enter the Input Coordinates as shown in the dialog box.
(c) Click Mark and close the Region Adaption dialog box.

4. Patch the following values in the reaction zone.
   ![Solution Initialization](patch_icon)
   ![Patch...](patch_icon)
   (a) Select cylinder-r0 in the Registers to Patch selection list and patch the following values:

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>2000</td>
</tr>
<tr>
<td>h2o</td>
<td>0.01</td>
</tr>
<tr>
<td>co2</td>
<td>0.01</td>
</tr>
</tbody>
</table>

   (b) Close the Patch dialog box.

5. Set the Under-Relaxation Factors as follows:
   ![Solution Controls](controls_icon)

<table>
<thead>
<tr>
<th>Category</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Species</td>
<td>0.95</td>
</tr>
<tr>
<td>Energy</td>
<td>0.95</td>
</tr>
<tr>
<td>Discrete Phase Sources</td>
<td>1</td>
</tr>
</tbody>
</table>

6. Request 1 iteration.
   ![Run Calculation](run_icon)

7. Save the case and data files (coal-ebu-react-start.cas.gz and coal-ebu-react-start.dat.gz).
   ![File](file_icon) Write Case & Data...
8. Change the Number of Continuous Phase Iterations per DPM Iteration to 25.
   ![Models → Discrete Phase → Edit...]

9. Change the Vaporization Temperature (k) to 773.
   ![Materials → coal-particle → Create/Edit]

10. Set the Under-Relaxation Factor for Discrete Phase Sources to 0.5.
    ![Solution Controls]

11. Request 500 more iterations.
    ![Run Calculation]

12. Save the case and data files (coal-ebu-react.cas.gz and coal-ebu-react.dat.gz).
    File → Write → Case & Data...

**Step 10: Reacting Flow Solution Including Radiation**

1. Change the solution control parameters.
   ![Solution Controls]
   (a) Click on Equations and select Discrete Ordinates from the list of Equations.
   (b) Click OK to close the equations dialog box.

2. Request 100 additional iterations.
   ![Run Calculation]

   File → Write → Case & Data...

4. Change the under-relaxation factors.
   ![Solution Controls]
   (a) Enter 1 for Species and Energy.

5. Request 500 additional iterations.
   ![Run Calculation]

   File → Write → Case & Data...
Coal Combustion with Eddy Break Up (EBU) Model

Step 11: Reacting Flow Solution Including Particle-Radiation Interaction

1. Enable Particle Radiation Interaction from the Physical Models tab in Discrete Phase Model dialog box.

\[ \overset{\text{Models}}{\Rightarrow} \text{Discrete Phase} \overset{\text{Edit...}}{\Rightarrow} \]

*Click OK in the Information dialog box.*

2. Ensure that Particle Emissivity and Particle Scattering Factor are set to 0.9.

\[ \overset{\text{Materials}}{\Rightarrow} \overset{\text{Combusting Particle}}{\Rightarrow} \text{Create/Edit...} \]

**Note:** It is better to use different values of emissivity for Char, Volatile, and ash. This can be done using a UDF for particle emissivity. `DEFINE_DPMPROPERTY` can be used for this purpose and a sample function is provided in the `emissivity.c` file. There are some macros in this UDF which will work only with compiled UDF, This tutorial is created with interpreted UDF. Therefore, varying emissivity is not used here.

3. Change the under-relaxation factors.

\[ \overset{\text{Solution Controls}}{\Rightarrow} \]

(a) Enter 0.25 for Discrete Phase Source.


\[ \overset{\text{Run Calculation}}{\Rightarrow} \text{Calculate} \]


Step 12: Postprocessing

1. Check the mass balance for convergence.

\[ \overset{\text{Reports}}{\Rightarrow} \overset{\text{Fluxes}}{\Rightarrow} \text{Set Up...} \]

(a) Select Mass Flow Rate in the Options list.

(b) Select all the zones from the Boundaries selection list and click Compute.

*This is net gas phase mass flux.*

2. Check the net heat transfer.

\[ \overset{\text{Reports}}{\Rightarrow} \overset{\text{Fluxes}}{\Rightarrow} \text{Set Up...} \]

(a) Select Total Heat Transfer Rate in the Options list.

(b) Select all the zones from the Boundaries selection list and click Compute.

*This is net gas phase heat transfer.*

3. Display filled contours of velocity magnitude (Figure 3) on \( x=0 \text{-plane} \).

4. Display filled contours of static temperature (Figure 4).
Coal Combustion with Eddy Break Up (EBU) Model

Figure 3: Contours of Velocity Magnitude

Figure 4: Contours of Static Temperature
5. Display filled contours of mass fraction of vol (Figure 5), \( \text{o}_2 \) (Figure 6), \( \text{co}_2 \) (Figure 7), \( \text{co} \) (Figure 8), and \( \text{h}_2\text{o} \) (Figure 9) on x=0m plane.

![Figure 5: Mass Fraction of Volatiles](image1)

![Figure 6: Mass Fraction of \( \text{o}_2 \)](image2)
Figure 7: Mass Fraction of $\text{CO}_2$

Figure 8: Mass Fraction of $\text{CO}$
6. Display DPM summary.

(a) Enter 1 for Number of Continuous Phase Iterations per DPM Iteration in the Discrete Phase Model dialog box and click OK.

(b) Enter 1 for Discrete Phase Sources under-relaxation factor.

(c) Display DPM summary by entering the following command in TUI.

```
/report>dpm-summary
```

Note: This will show the conversion of char, volatile and moisture. This should be close to 100%.
Figure 10: DPM Summary

Step 13: NOx Modeling

1. Select the NOx model.

   (a) Enable Thermal NOx, Prompt NOx, and Fuel NOx from the Pathways list.

   (b) Select vol from the Fuel Species selection list.


   (d) In the Prompt tab, enter 2.8 and 0.685 for Fuel Carbon Number and Equivalence Ratio respectively.

   (e) Click the Fuel tab and select Solid from the Fuel Type group box.

      i. Set the following parameters:
Coal Combustion with Eddy Break Up (EBU) Model

<table>
<thead>
<tr>
<th>Fuel Type</th>
<th>Solid</th>
</tr>
</thead>
<tbody>
<tr>
<td>N Intermediate</td>
<td>hcn/nh3/no</td>
</tr>
<tr>
<td>Volatile N Mass Fraction</td>
<td>0.009091</td>
</tr>
<tr>
<td>Partition Fractions</td>
<td></td>
</tr>
<tr>
<td>hcn</td>
<td>0.9</td>
</tr>
<tr>
<td>nh3</td>
<td>0.1</td>
</tr>
<tr>
<td>Char N Conversion</td>
<td>no</td>
</tr>
<tr>
<td>Char N Mass Fraction</td>
<td>0.041176</td>
</tr>
</tbody>
</table>

The values of volatile and char N mass fractions are obtained from Step 3.4, where you set the Species Transport Model.

**Note:** Refer to Section 23.1.1.7.2 Setting Solid(Coal) Fuel NOx Parameters in the ANSYS FLUENT User’s Guide for details on how to calculate volatile and char N mass fractions.

(f) Click the Turbulence Interaction tab and select temperature from the PDF Mode drop-down list.

(g) Enter 20 for PDF Points.

(h) Click Apply and close the NOx Model dialog box.

2. Set Number of Continuous Phase Iterations per DPM Iteration to 0 in the Discrete Phase Model and click OK.

3. Change the solution control parameters.

   ![Models](Models) Discrete Phase → Edit...

4. Request for 100 iterations.

5. Save the case and data files (coal-ebu-final-no.cas.gz and coal-ebu-final-no.dat.gz).

6. Display contours of mass fraction of Pollutant no on x=0m plane (Figure 11).

   ![Graphics and Animations](Graphics and Animations) Contours → Set Up...

   (a) Select NOx... and Mass Fraction of Pollutant no from the Contours of drop-down lists.

   (b) Select x=0m plane from the Surfaces selection list.

   (c) Click Display and close the Contours dialog box.

© ANSYS, Inc. December 5, 2012
Summary

Application of the EBU model in a coal combustion case has been demonstrated.

In this tutorial, nine injections are introduced at the inlet. The coal-hv particles travel a short distance before they start releasing volatiles. At this point, reactions start and the temperature increases. The high temperature zone is seen inside the furnace slightly away from the inlet. In Eddy Break Up (EBU) coal combustion, coal particles release volatiles that react with oxygen and produce combustion products. Similar trends can be seen for NOx.

The stoichiometric coefficients can be calculated once chemical composition of coal volatiles is known.

For more information on determining coal volatile composition, refer to the ANSYS FLUENT 14.5 Tutorial Guide.